EEC 134 Radar System Design

Application Note: PCB Design with KiCad

Team RF Eater

Qun Xia

I. Abstract

For the radar system design project, printed circuit board (aka. PCB) can make the whole system simpler and lighter. There is also less chance to have a short or broken circuit problem inside a PCB. In this two-quarters senior design project, my main responsibility in our group was to design the PCB used in our system. In this application note, I will discuss the procedures and tips of PCB design based on personal experience.

II. Introduction

I had no PCB design experience before starting this senior design, so I used the software KiCad, recommended by Professor Liu, to design PCB. The PCB our team designed and assembled in fall quarter were all failed, so in winter quarter we decided to design three smaller PCB instead of a large one for the baseband circuit, which made the debugging easier. It is also more economical; we do not need to replace the whole baseband if there is a problem with a certain part of the circuit. We totally designed four PCBs for our radar system, one for each of the following: function generator, voltage regulator and reference, active low-pass filter with gain stage and low-noise filter.

III. Using KiCad

It is important to get familiar with the software and know how to use each function before starting your design, especially if you are new to the PCB design like me. The PCB tutorial document provided by Professor Liu is helpful. It will be a good start to read and follow the steps given in the tutorial.

IV. Schematic Design

The first step of PCB design is schematic design. It will be a good idea to draw the schematic of the circuit on paper first rather than directly drawing it in the software. Mark each component clear with their name and values. Then figure out what components and how many input and output pins needed for your circuit.

When you do the schematic design with the software, you will need to create your own schematic symbols if there is no proper schematic symbol in the default KiCad library for the component that you need. It is important to match each pin number and pin name correctly. You can check the schematic symbol with the datasheet for each component you draw. Fig 1 is an example of the schematic symbol I designed for LT1009:

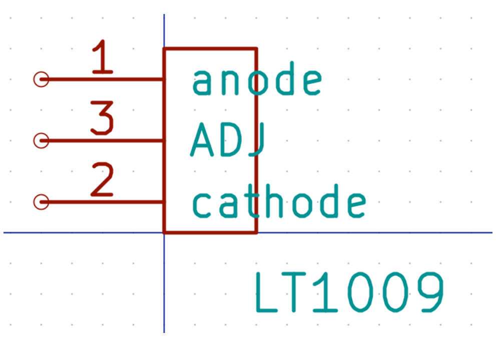


Fig 1. LT1009 Schematic Symbol

After having all the schematic symbol you need, you can start drawing the whole circuit and connecting them. When connecting the circuit, do not connect them with segments directly. You can connect the components by labeling the segments, like what I did in Fig 2:

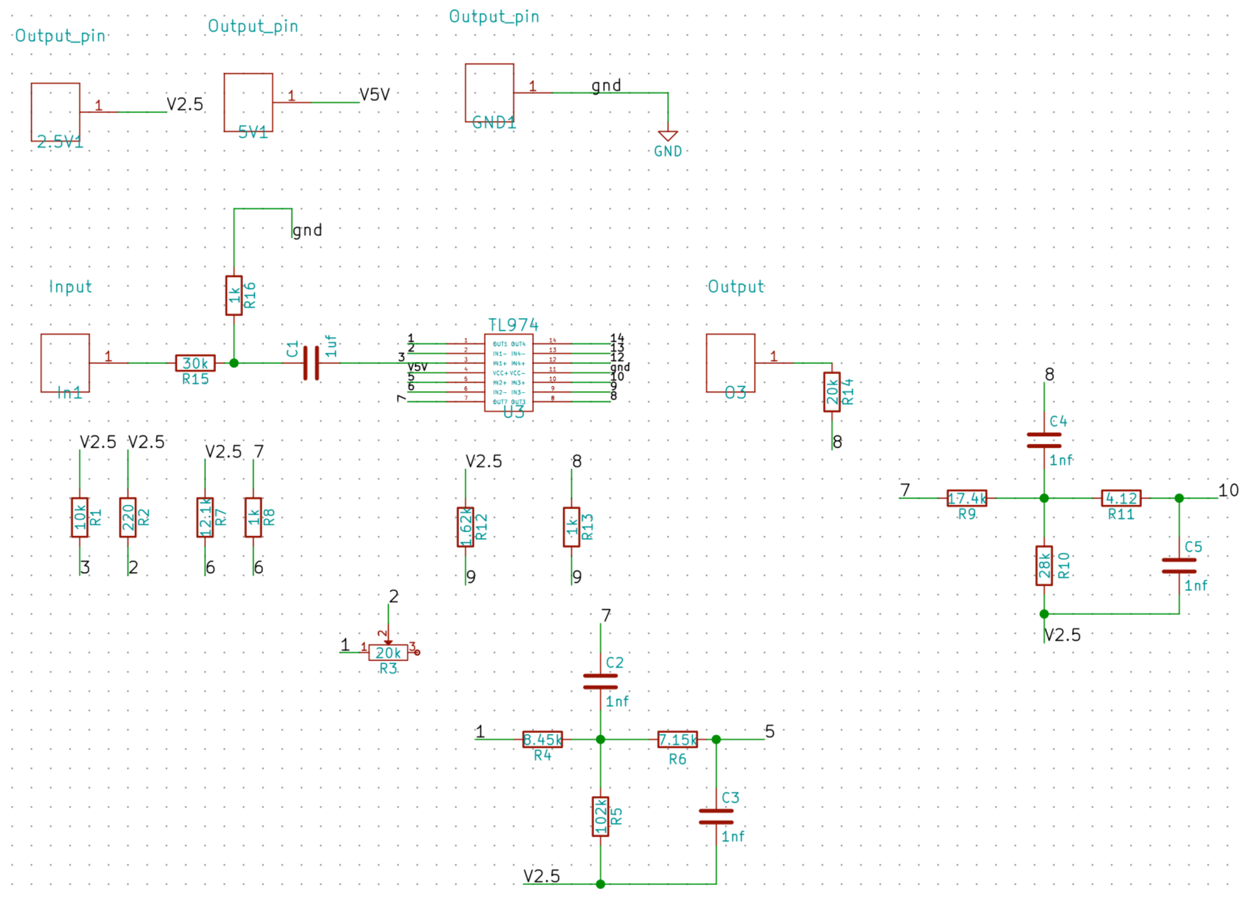


Fig 2. LPF + Gain State Schematic

When you finish drawing the schematic, do not hurry to move on to the next step. First check if you have labeled all the components correctly. Label all the components with proper reference number and correct values can make everything easier when assembling the PCB. Secondly, check if you add enough input, output and test pins. Thirdly, perform an electrical rule check by clicking the icon shown in Fig 3, which you can find at the top bar of your schematic window. The electrical rules check can check the connections between components for you.

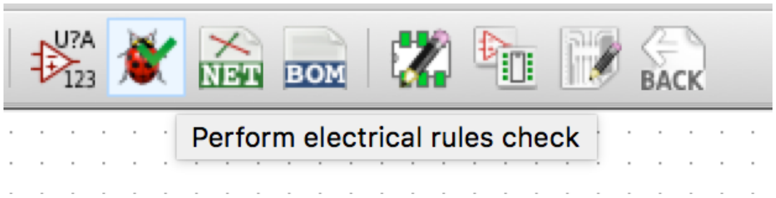


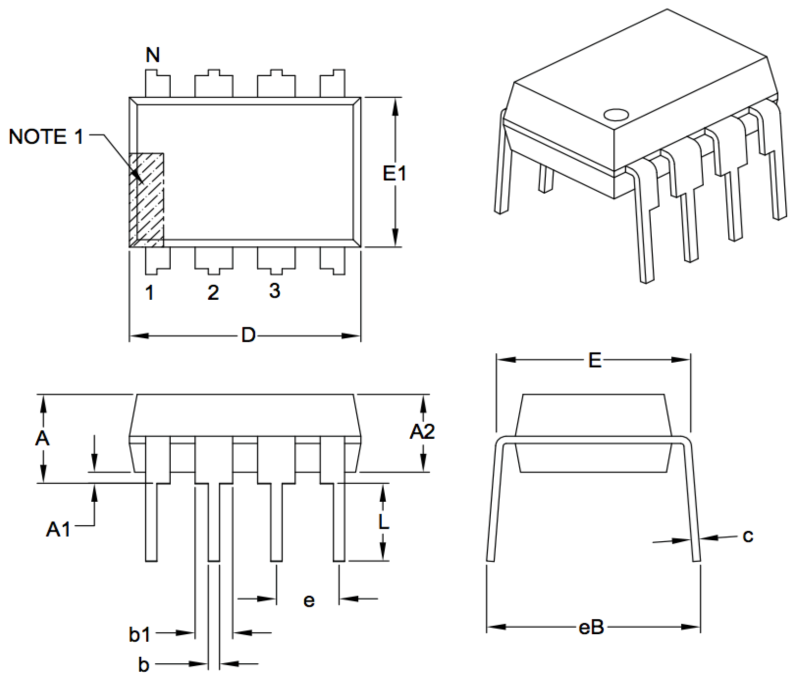
Fig 3. Electrical Rules Check

V. Creating Footprint

After finishing the schematic design, you are now moving on to an important step, to assign component footprint. The footprint is the actual look of your component. For the capacitors and resistors, you should use SMD 0603, for which the footprint is available in the library. For most components, you may not find the footprint in the library, so you will need to create your own footprints. Since the footprint will be the real outline of the component fabricated on the PCB, any error with the footprint can cause a problem during assembling.

First draw a general outline of your component with proper pins in the footprint editor. Then set the parameters with actual size. You can get the parameter data from datasheet. Fig 4 is an example of the parameter data for MCP4921 from datasheet. If you cannot find the parameter from datasheet, you can measure it with a ruler. It will be a good idea to always compare the data with your own measurement even though the parameters are given in the datasheet.

Besides size, another important thing for footprint design is the pin number and pin name. The outline you draw in footprint editor is a top view of your component, so make sure you assign the pin number correctly. The pin number should match the pin number you used for schematic design. Fig 5 shows an example of the LT1009 footprint I designed, which has an error for the pin number. Thanks to the structure of LT1009, I fixed this problem during assembling. Do not forget to generate the netlist file for your schematic.



		Units	INCHES		
		Dimension Limits	MIN	NOM	MAX
Number of Pins	N		8		
Pitch	e		.100 BSC		
Top to Seating Plane	A		–	–	.210
Molded Package Thickness	A2		.115	.130	.195
Base to Seating Plane	A1		.015	–	–
Shoulder to Shoulder Width	E		.290	.310	.325
Molded Package Width	E1		.240	.250	.280
Overall Length	D		.348	.365	.400
Tip to Seating Plane	L		.115	.130	.150
Lead Thickness	c		.008	.010	.015
Upper Lead Width	b1		.040	.060	.070
Lower Lead Width	b		.014	.018	.022
Overall Row Spacing §	eB		–	–	.430

Fig 4. MCP 4921 Parameters

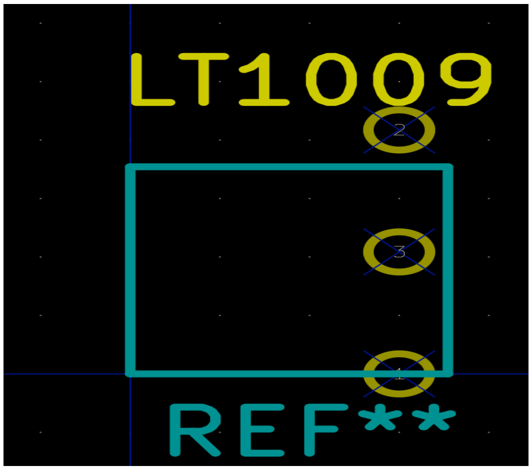


Fig 5. LT1009 Footprint with Error

VI. Lay Out of PCB Design

PCB lay out design is also an important step. This step will decide how your actual PCB looks like. This is also hardest step, so you need to be patient with this step. First draw a rectangular on the Edge.Cuts layer. You can start with a 2x2 inches rectangular, and adjust the size when you finish the lay out design. Read the netlist and then all the components will be shown on the screen. Arrange the large components first. You can refer to your circuit on the breadboard if you have it and get a general idea of how the whole circuit will look like. Check the design rules from Bay Area Circuit website and set up the trace width. Then drag the components to the position you want and connect the pins as you designed in your schematic. Since you will fill zone of the PCB with ground, you do not need to connect ground pins. Fig 6 shows an example of the PCB lay out for our baseband filter and gain stage, which is 1.5x1.5 inches. Put the input/output pins near the edge; this will make it easier for you to connect the pins with pins from other board. Also, remember to leave a reasonable space between each pin; this can prevent circuit shorting when connect pins with power supply in future use. Adjust the size of the edge cuts to make it just large enough to include all the components. If the PCB has too much empty space, it can have significant power and signal loss.

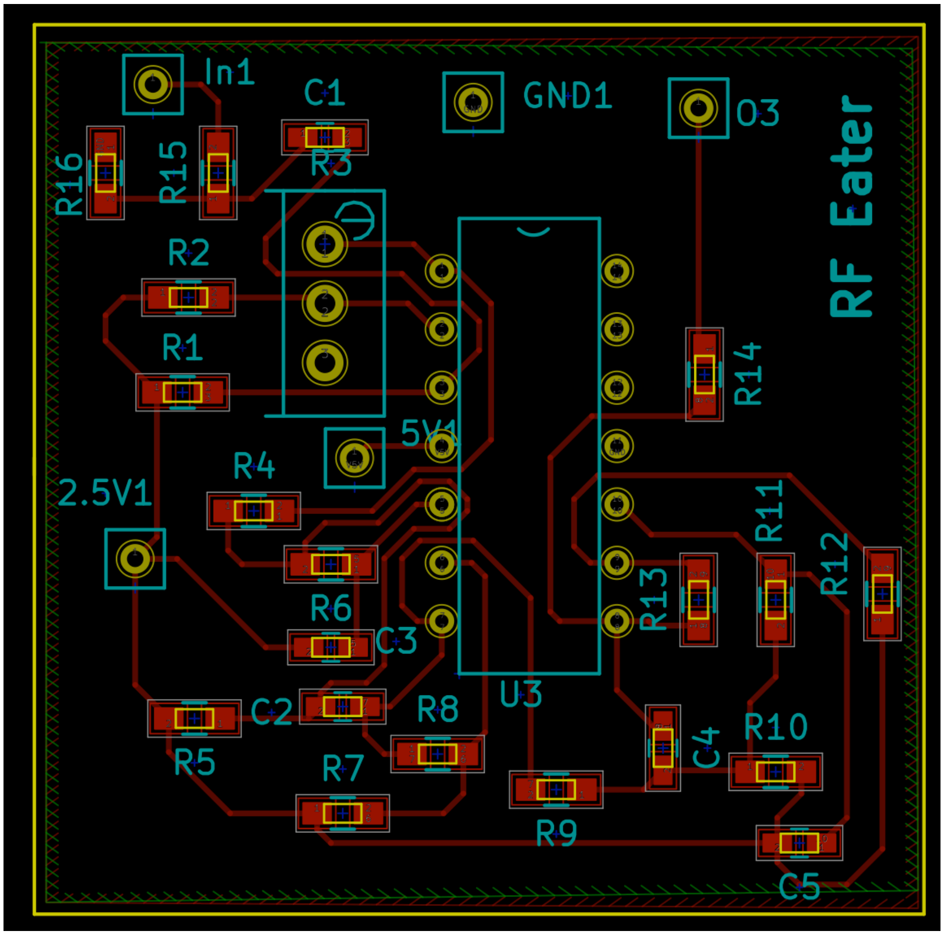


Fig 6. LPF + Gain Stage PCB Lay Out

The PCB Lay Out Design for RF components will need some extra steps, especially when they need to be operated in high frequency. Fig 7 shows the PCB lay out design for our low-noise amplifier. You will need 50ohm transmission line between SMA connector and the IC due to the impedance matching in high frequency. You can achieve this by making the copper trace between SMA connector and IC thicker than other trace. Another extra step is to add via fence to PCB. You can achieve this by adding a lot of through holes or vias around the edge of the PCB and also around the IC. This step can help with decreasing the inductance in the PCB under high frequency.

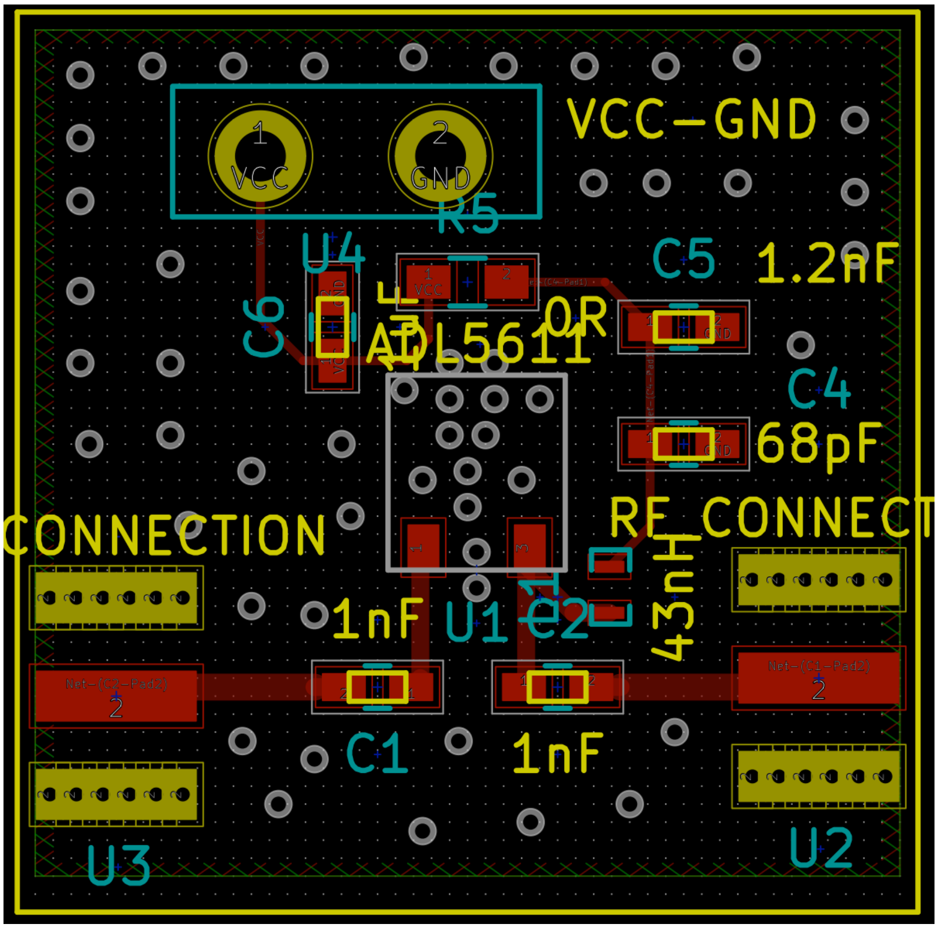


Fig 7. LNA PCB Lay Out

After finishing the PCB design, fill the zone of PCB with ground. Remember to perform the design rules check before you generate the gerber and drill files. This will help you check the connection between components and also the size of the trace and holes. You should also let someone else review your whole PCB design to make sure you do not make any mistake due to careless.

VII. DFM Report

Submit the gerber and drill files to the Bay Area Circuit website to get a DFM Report. If there is any error shown in the DFM report, go back to your design and correct them. The DFM report will tell you what is the problem with. Based on my own experience, most errors happened with the drill size or plate size with the through holes. The DFM report usually does not tell you too much about the details for the error. You can always email Bay Area Circuit to get more details about the errors. The technician from BAC can check you design again and offer you more details. They usually reply fast.

VIII. Conclusions

PCB design is a useful skill for electrical engineer. You will need it for a lot kind of jobs in electrical engineering career field. EEC 134 project is a good chance for you to learn how to do PCB design from a beginner.